

Openscad to cnc mill

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straygrey

Posts: 33

Joined: Tue Oct 05, 2021 12:14 pm

Location: South Africa

Contact: ...

Openscad to cnc mill

by straygrey » Sun Apr 16, 2023 1:01 pm

Being a long time programmer I find openscad a convenient editing platform and have therefore created

Davidaphotoboxcnc.scad

(735 Bytes) Downloaded 76 times

I have successfully 3d printed this file but now wish to cut it from a piece of wood. So far I have

Davidaphotoblockcnc.fcstd

(35.54 KiB) Downloaded 79 times

but I am somewhat lost and do not know what my next step should be.

Please advise.

OS: Debian GNU/Linux 11 (bullseye) (GNOME/gnome)

Locale: English/South Africa (en_ZA)

lgrfbs

Posts: 142

Joined: Wed Apr 07, 2021 9:51 am

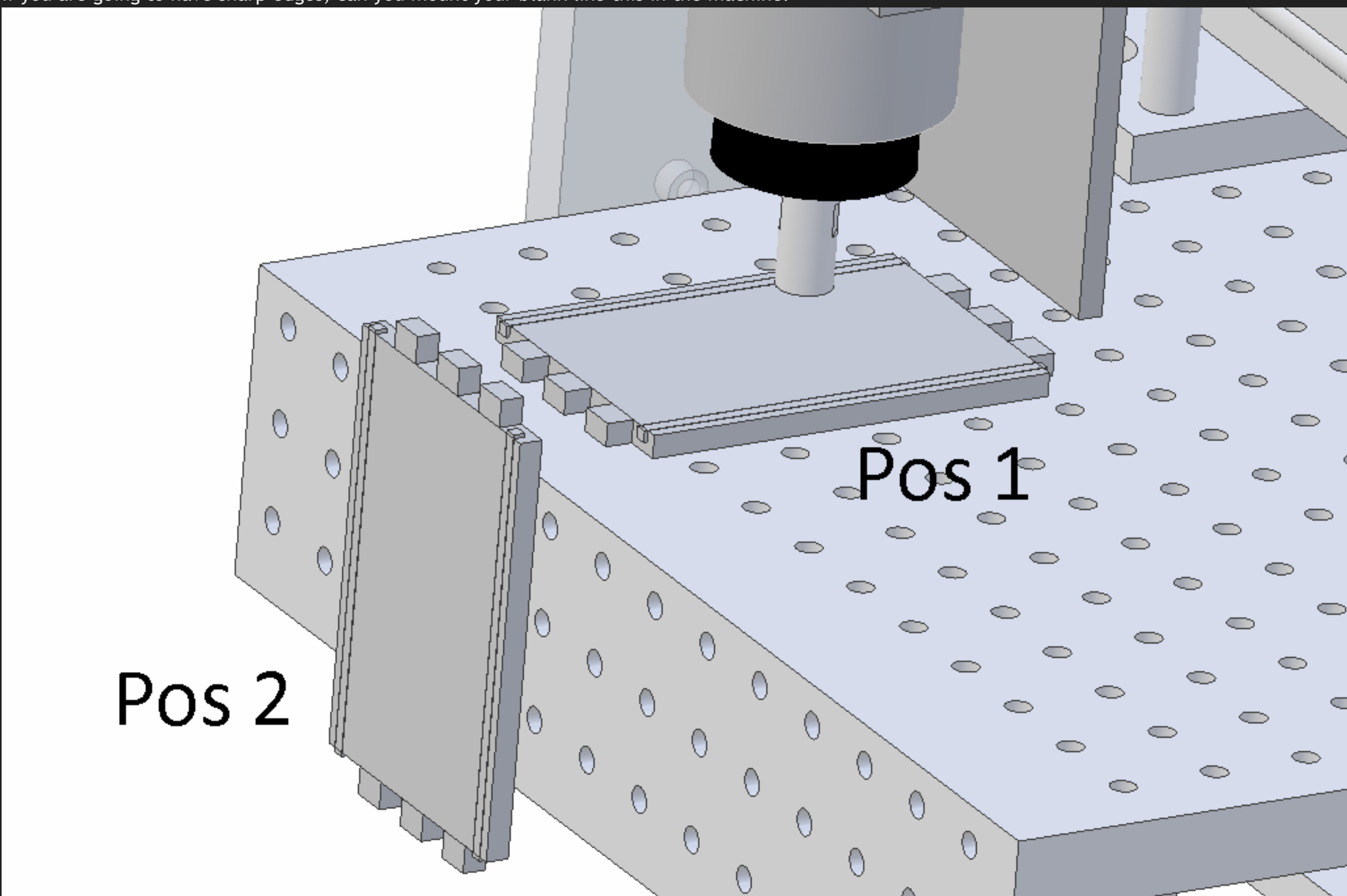
Location: Sweden

Contact: ...

Re: Openscad to cnc mill

by lgrfbs » Sun Apr 16, 2023 2:11 pm

If you are going to have sharp edges, can you mount your blank like this in the machine?



Rigg.png (56.35 KiB) Viewed 1666 times

If you have to run with a endmill, what is the minimum radius your part can work with?

Windows 10 64bit

Best FreeCAD for me right now: FreeCAD 0.18.4 and testing FreeCAD 0.21.1 + Better Tool Library (BTL) = Now I can use FreeCAD.

Main CAD program: Alibre Design™ Expert, Version: V25 64-bit [Build 25033]

straygrey

Posts: 33

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Contact: ...

Re: Openscad to cnc mill

by straygrey » Sun Apr 16, 2023 2:42 pm

I do not have a problem cleaning up the corners with a manual chisel after cnc cutting. Mounting the blank vertically would be an issue for then one would have to cut the slots by re mounting the blank. This would be my 1st mill cut with Freecad and once I get that behind me I will be happier.

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straygrey

Posts: 33

Joined: Tue Oct 05, 2021 12:14 pm

Location: South Africa

Contact: ...

Re: Openscad to cnc mill

by straygrey » Sun Apr 16, 2023 2:47 pm

Of course I could also do a dogbone. Will look at that when I learn more.

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lgrfbs

Posts: 142

Joined: Wed Apr 07, 2021 9:51 am

Location: Sweden

Contact: ...

Re: Openscad to cnc mill

by lgrfbs » Sun Apr 16, 2023 2:57 pm

Can I guess that you already have a CNC machine and cuting tools?

Youtube keywords: "FreeCAD Path Milling", so you can find newer videos that might suit you better.

One I lake (3 Year old):

FreeCAD- The Powerful Path Workbench for CNC Maching and Gcode| JOKO ENGINEERING|

<https://youtu.be/MWFC17MlFOE>

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straygrey

Posts: 33

Joined: Tue Oct 05, 2021 12:14 pm

Location: South Africa

Contact: ...

Re: Openscad to cnc mill

by straygrey » Sun Apr 16, 2023 3:20 pm

Yes I do have a 3018 machine and tools.

I perceive that I have a number of Jobs to undertake as follows

1) A job to cut each of the 3 slots. ie 1 job per slot

2) A job to cut the complete side out of the base.

My first issue is that when I select the 1st slot by clicking on it in the Combo view Model and expect to be able to create a job by clicking on the JobCreate icon but cannot as the job create icon is greyed out. What am I misunderstanding or what did I do incorrectly?

OS: Debian GNU/Linux 11 (bullseye) (GNOME/gnome)

Locale: English/South Africa (en_ZA)

lgrfbs

Posts: 142

Joined: Wed Apr 07, 2021 9:51 am

Location: Sweden

Contact: ...

Re: Openscad to cnc mill

by lgrfbs » Sun Apr 16, 2023 4:07 pm

Which version of FreeCAD do you have?

I don't have much more to say, so you'll have to rely on the other professionals on this forum.

Just because I am curious which tool will you use?

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straygrey

Posts: 33

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Contact: ...

Re: Openscad to cnc mill

by straygrey » Sun Apr 16, 2023 4:55 pm

See my signature. I know that I am not supposed to put the Freecad version in the signature but until I learn a better way this will have to do.

Code: [Select all](#)

```
OS: Debian GNU/Linux 11 (bullseye) (GNOME/gnome)
Word size of FreeCAD: 64-bit
Version: 0.20.1.29410 (Git) AppImage
Build type: Release
Branch: (HEAD detached at 0.20.1)
Hash: f5d13554ecc7a456fb6e970568ae5c74ba727563
Python 3.10.5, Qt 5.15.4, Coin 4.0.0, Vtk 9.1.0, OCC 7.6.2
Locale: English/South Africa (en_ZA)
Installed mods:
 * Help 1.0.3
```

OS: Debian GNU/Linux 11 (bullseye) (GNOME/gnome)

Locale: English/South Africa (en_ZA)

Paula

Posts: 51

Joined: Tue Jan 26, 2021 10:06 pm

Contact: ...

Re: Openscad to cnc mill

by Paula » Sun Apr 16, 2023 5:34 pm

Hello,

I've just looked at your FreeCAD file. I think you only need one Job but will have several operations within that job. The first thing to do though is make a single object from all the objects under "Group". You can do this in the Part workbench. This will create an object called "Fusion". Switch to the Path workbench, select "Fusion" and then click the Create Job icon to create the job. You should be able to do all the needed operations with a 3.175mm(1/8th inch) end mill. I'll not go into detail about all the things you'll need to do but will instead advise that you start with a simpler test object and learn the ins and outs of pocketing and profiling. CNC has quite a steep learning curve and unlike 3D printing you don't get complexity for free. Hope this helps.

GeneFC

FreeCAD Guru

Posts: 5373

Joined: Sat Mar 19, 2016 3:36 pm

Location: Punta Gorda, FL

Contact: ...

Re: Openscad to cnc mill

by GeneFC » Sun Apr 16, 2023 7:53 pm

“straygrey wrote: Sun Apr 16, 2023 3:20 pm

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This is not the preferred workflow. It would seem you may not have read the introductory material for the Path WB.

I recommend looking at the bottom of the wiki page https://wiki.freecad.org/Path_Workbench and take a look at the [https://wiki.freecad.org/Path_Walkthrou ... _Impatient](https://wiki.freecad.org/Path_Walkthrou..._Impatient) and perhaps some of the tutorials.

Gene

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